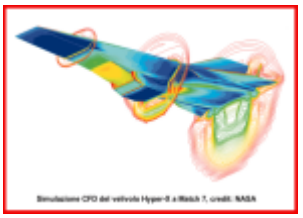

CAPI 2007
Calcolo ad Alte Prestazioni in Italia
Fluidodinamica computazionale
Milano, 15-16 Ottobre 2007

http://www.cilea.it/capi2007/

Titoli ed abstracts degli interventi

Keywords: Supercalcolo, Fluidodinamica.

Lunedì 15 Ottobre 2007
Prima Sessione (Automotive) - CFD per la Progettazione industriale

CFD simulations for internal combustion engines: combustion, internal flows, integrated 1D-multiD simulations.

Gianluca D'Errico - Dipartimento di Energetica, Politecnico di Milano
Tommaso Lucchini - Dipartimento di Energetica, Politecnico di Milano
Gianluca Montenegro - Dipartimento di Energetica, Politecnico di Milano

Computational fluid dynamics (CFD) is widely applied to design and optimise the internal combustion engine components. In this way time and costs for experiments and prototyping are strongly reduced since only the most promising configurations are tested. However, I.C. engine simulation represents a very challenging task for CFD modelling and for this reason the number of CFD codes capable to perform a successful flow simulation in IC engine is very small. Reliable computational tools are demanded to handle multiple interacting thermo-fluid processes and geometrical constraints imposed by the engine geometry.

Generally high grid resolutions are used, to correctly capture the main physical phenomena taking place within the engine cylinder, intake and exhaust systems but this seriously increase the computational time. To reduce it, the CFD tool must support parallel computation.

In this work, the authors intend to present several applications of the OpenFOAM code in the field of I.C. engine simulations.

The code supports unstructured, polyhedral meshes of arbitrary shapes and can be used for the simulation of any kind of complex geometries. In particular, it includes libraries for combustion, turbulence and sprays which have been extensively validated. Finally, its C++, object-oriented structure allows easy implementation of any new physical and numerical model, re-using available software components like sparse matrix class, linear solvers, field algebra implementation and others. The software design transparently supports parallel execution without any programming effort in the top-level code where processor-to-processor communications are handled as a type of coupled boundaries.

OpenFOAM currently relies on the public domain MPI implementation, LAM, which has proven to give good performance and be extremely stable.

In this specific study, the OpenFOAM code has been used for three different purposes: prediction of the flow coefficients for an engine cylinder head, simulation of in-cylinder flow and combustion and 1D-multiD full cycle coupled simulations for complex geometries inside the intake and exhaust systems. A good matching between the calculated and measured results was achieved for all the tested configurations, pointing out that the computational runtime fairly scales with the number of processors.

Analisi Numerica Fluido-Struttura del collettore di scarico di un motore turbo-compresso GDI.

Tarcisio Colombo - FIAT POWERTRAIN TECHNOLOGIES, Gasoline Engine Design Arese
Angelo Rosetti - FIAT POWERTRAIN TECHNOLOGIES, Gasoline Engine Design Arese

Lo studio svolto riguarda il collettore di scarico di un motore a combustione interna 4 cilindri GDI turbo-compresso con lo scopo di valutare contemporaneamente la fluidodinamica all'interno del collettore e le sollecitazioni strutturali dovute alla temperatura. E' stato valutato l'effetto di differenti mesh ed il calcolo è stato svolto in diverse modalità: stazionario, transitorio ed un calcolo combinato CFD 1d-3d per simulare l'effettivo comportamento del componente accoppiato al motore. Le analisi sono state effettuate su un Mini-Cluster di calcolo composto da una macchina Linux e 2 macchine WMWare "virtuali" ed i software utilizzati sono StarCD (CD-Adapco) e GT-Power (Gamma Technologies).

I risultati sono stati confrontati con dati sperimentali e calcoli strutturali effettuati con codici agli elementi finiti.

CFD analysis for the ballast projection on high speed trains.

Federico Cheli - Dipartimento di Meccanica, Politecnico di Milano
Francesco Ripamonti - Dipartimento di Meccanica, Politecnico di Milano
Daniele Rocchi - Dipartimento di Meccanica, Politecnico di Milano
Ignazio Viola - Dipartimento di Meccanica, Politecnico di Milano

The ballast projection is a problem that occurs on the high speed train lines when the transit of a convoy makes the stones spring up from the track bed. Once a stone is released from the ballast, its interaction with the flow field underneath the train may lead the stone to be thrown against the train wall causing some impacts that may damage some parts of the vehicle. In particular, some auxiliary parts, that are usually located under the train may be affected by the problem and some maintenance extra costs arise. To better understand the phenomenon, a cooperation among the train producers and the line managers was started up to investigate the flow condition under the train both by an experimental and by a numerical point of view. The problem requires the analysis of several aspects, starting from the influence of the ballast characteristic on the problem, to the train geometry effects.

Some full scale tests have been performed to measure the velocity fields in some points along the line during the train passing. Wind tunnel tests on a ballast track bed have been performed and some mechanical test to define a model of the flying stone are still on going. The Mechanical Department of the Politecnico di Milano participates to the research activity, using the CFD approach to simulate the flow field conditions occurring under the train for different line scenarios. The aim of the research is to define the flow patterns under the train vehicles with particular attention to the bogie cavities and to the intercar gap regions that are the parts of the train where the ballast projection cause the more severe problems. The work, that is still on going, consists in simulating the flow field under the different vehicles of the convoy by using a marching technique. In fact, due to the large amount of cells that would be requested for meshing a whole domain containing the train and the line, it has been decided to split the domain in subparts containing up to a whole vehicle and to use the simulation results on the previous part as input flow conditions for the following part. The results of this activity will be compared to measurements performed on the line and will be used as boundary conditions for a numerical simulation of the stone dynamics.

The project aims to evaluate the influence of the different parameters on the problem to find operative solutions to be implemented in the new lines and train construction but also in the existing lines.

Seconda Sessione (Naval and Aerospace)

Xnavis: a parallel CFD code for the numerical simulations of unsteady free surface flows for naval hydrodynamics.

Giorgio Amati - CASPUR
Riccardo Broglia - INSEAN

Simulations of flows around hulls are really CPU time expensive: here a Parallel version of a CFD code will be presented.

Xinavis is based on the finite volume discretization of the unsteady incompressible Navier-Stokes equations. The free surface is handled by means of the unsteady one-phase level set approach. Dynamical multi-block overlapping grids are used for the discretization of the physical domain.
The code has been used for the analysis of various fluidodynamics related problems in the framework of both naval and aeronautical context.

Example of simulations range from the full appended surface piercing hulls in steady motion or with either prescribed or predicted manoeuvres in both calm water andi n waves situations; analysis of the turbulent flow around submarines;

CFD study of the flow around a plane for the evaluation of the radiated noise.

This code present many challenging features for High Performance Computing, e.g Load Balancing, complex geometries, bodies in relative motion or around a body with a prescribed/predicted motion and so on.

To face all these aspects a "grain/fine" parallelization of the unsteady RANS
code has been achieved by distributing the structured blocks among the available non-shared or shared memory processors, and by spreading the computational work to be done inside each block among available shared memory processors.

Communication between processors for the "grain" parallelization is obtained by using standard Message Passing Interface (MPI) library, whereas "fine" (shared memory) parallelization is achieved by means of Open Message Passing (OpenMP) library.

CFD Calculation of the Hydrodynamic Characteristics of Planing Hulls.

Stefano Brizzolara - Dipartimento di Ingegneria navale e Tecnologie marine, Università degli Studi di Genova
Davide Grassi - Dipartimento di Ingegneria navale e Tecnologie marine, Università degli Studi di Genova
Giovanna Vatteroni - Dipartimento di Ingegneria navale e Tecnologie marine, Università degli Studi di Genova
Diego Villa - Dipartimento di Ingegneria navale e Tecnologie marine, Università degli Studi di Genova

Paper presents the main results achieved by the hydrodynamic CFD group of the University of Genova for the CFD modeling of the hydrodynamic characteristics of planing hulls. The commercial software Starccm+ has been used for the presented studies which are currently ongoing and under continuous evolution. The hydrodynamic problem of the external flow on the hull is of course complicated by the quite rapid evolution of the free surface along the hull, with the formation of forward jet spray and a highly curved free surface shape at bow, the large pressure gradients in the stagnation area on the bottom, the sharp flow separation along the chines downstream, and the formation of the wavy free surface at stern. The ability of the code to accurately define all these physical phenomena has been tested and validated in the studies. Numerical results are presented and discussed in the paper in terms of distributed hydrodynamic parameters, such as wave heights, pressure and wall strain rates distribution along the hull and where possible compared with available experimental results on standard hull shapes taken from systematic model series of experiments in
towing tanks. Global hydrodynamic parameters such as the lift, drag and moment coefficients numerically predicted by the RANSE solver for the hull at given speed and fixed dynamic attitude are compared with the available measurements done for scaled models tested in towing tank.

Particular attention is also given to the solution of the steady dynamic attitude (trim and sink) prediction through the numerical simulations. In fact, in case the hull is left free to move in the longitudinal plane, with respect to the initial static attitude (at rest), it will experience a different dynamic attitude at high speeds, very much depending on the running speed. The lift trim moment and finally induced resistance experienced by the hull are largely dependent from the dynamic attitude which in turn are influenced by the hull shape, so the accurate prediction of the global forces and

moments at high speed is essential for the design of the hull. The validation of the results show a very good correlation with available experimental results, at least for engineering purposes, in a field (that of planing hulls) where at present the design practice relies mainly in expensive and time consuming model tests or very approximate prediction methods based on simplified semi-empirical theories.

Computational Aeroelasticity with CFD models.
Luca Cavagna - Dipartimento di Ingegneria Aerospaziale, Politecnico di Milano

This paper gives an overview of the numerical procedures which are nowadays used at Dipartimento di Ingegneria Aerospaziale Politecnico di Milano for computational aeroelasticity by means of omplex Computational

Fluid Dynamics (CFD) models. Two main research programs are here outlined: the design of a flutter suppression system for the forward swept wing of the in-house aeroelastic demonstrator X-DIA, and the transonic flutter assessment and static aeroelasticity analyses for high-speed aircrafts (transonic and supersonic). Both of them require an accurate and complex fluid models based on Euler or Navier-Stokes equations, to correctly investigate Fluid Structure Interaction (FSI) phenomena. For the first case, an accurate description of the pressure load and the hinge-line moments are required to correctly define control surfaces effectiveness and related viscous effects; for the second case the adoption it is necessary to investigate shock-wave influence on flutter mechanisms and pressure-field to overcome the lack of the classic linearized potential theories in this speed-regime. This article introduces an aeroelastic procedure for (FSI) problems modelling, based on the integration of different commercial software. Far more computational resources than those used in the classic approach are required to solve the dynamic equations of the computational fluid dynamics on deforming meshes with several thousands of elements. Therefore, particular attention is paid in order to define a strategy making the whole process efficient and well-suited for the realistic industrial environment problems. Current hardware resources, combined with the availability of specific software for structural and fluid dynamics analysis make the time mature for extending Computational Aeroelasticity (CA) beyond the academic research environment, without using a specifically developed code.

The procedure of analysis used here is based on a user-defined plug-in named NAEMO-CFD (Numerical AeroElastic MOdeller with CFD models) running under the CFD solver FLUENT which is enhanced with the capability of carrying out different kind of static and dynamic aeroelastic computations. The structural model is represented by its vibration modes further enhanced with static brach modes defined by the user; in this case MSC-NASTRAN solver is used as source of these data.

Martedì 16 Ottobre 2007

Prima Sessione - CFD per l'Energia e l'Ambiente: modelli ed applicazioni

Mesoscopic numerical modeling of reactive mixture flows in solid oxide fuel cells by lattice boltzamann and high performance parallel computing.
Pietro Asinari – Dipartimento di Energetica, Politecnico di Torino
Romano Borchiellini - Dipartimento di Energetica, Politecnico di Torino
Michele Calì - Dipartimento di Energetica, Politecnico di Torino

Solid oxide fuel cells (SOFCs) are receiving considerable interest since they are suited for both stationary and vehicle applications. The reduction of activation polarization, the elimination of expensive catalysts, potential integration with cogeneration systems, and the possibility of being able to consider different composition of syngas as the fuel are interesting technical challenges. The Department of Energy (DOE) of the United States a few years ago initiated a set of research projects (SECA - Solid State Energy Conversion Alliance) with the purpose of increasing the power density, reducing the manufacturing costs, and encouraging commercially cost-effective prototypes. The European Union, through the European Hydrogen and Fuel Cell Technology Platform: Strategic Research Agenda (January 2005), has indicated that the SOFC is a priority choice for stationary applications.

Mathematical models that predict performance can aid in the understanding and development of solid oxide fuel cells (SOFCs). Of course, various modeling approaches exist involving different length scales. In particular, very significant advances are now taking place using microscopic models to understand the complex composite structures of electrodes and three-phase boundaries. Ultimately these advances

should lead to predictions of cell behavior, which at present are measured empirically and inserted into macroscopic cell models.

In order to achieve this ambitious goal, the key idea is to numerically simulate the underlying microscopic phenomena in an effort to bring the mathematical description nearer to actual reality. In particular, some recently developed mesoscopic tools appear to be very promising since the microscopic approach is, in this particular case, partially included in the numerical method itself. In particular, the models based on the lattice Boltzmann method (LBM) treat the problem by reproducing the collisions among particles of the same type, among particles belonging to different species, and finally among the species and the solid obstructions.

Recently, a model developed by the authors was proposed which, based on LBM, models the fluid flow of reactive mixtures in randomly generated porous media by simulating the actual coupling interaction among the species. A parallel three–dimensional numerical code was developed in order to implement this model and to simulate the actual microscopic structures of SOFC porous electrodes. The code has been developed in C++ and a free communication library has been adopted (MPICH 1.3) based on MPI technology. The reported numerical results were obtained on two cluster facilities. The first one is System X at Virginia Tech (VT). It is essentially a computational platform made of 1100 dual-processor Apple XServe G5 (2200 total CPUs, each characterized by 2.3 GHz, 4 GB RAM e 80 GB HD), connected by Cisco Gigabit Ethernet and Mellanox switches. The second facility has been recently developed at "Politecnico di Torino" and it is made of 100 Pentium-4 nodes (each characterized by 2.8 GHz, 512 MB RAM and 40 GB HD).

Automi cellulari paralleli ed asincroni per la stima delle perdite idrologiche.

Giuseppe Mendicino - Dipartimento Difesa del Suolo, Università della Calabria
Alfonso Senatore - Dipartimento Difesa del Suolo, Università della Calabria
Giandomenico Spezzano - Consiglio Nazionale delle Ricerche, Istituto di Calcolo e Reti ad Alte Prestazioni
Salvatore Straface - Dipartimento Difesa del Suolo, Università della Calabria
Pasquale Versace - Dipartimento Difesa del Suolo, Università della Calabria

La nota descrive un modello tridimensionale per la simulazione dell'infiltrazione dell'acqua nel suolo, sviluppato in un ambiente di calcolo parallelo basato su un'estensione dell'originale paradigma degli automi cellulari (AC). Il modello, ideato per la simulazione di sistemi su grande scala, adotta un approccio macroscopico dove le interazioni locali tra gli automi sono governate da leggi dal chiaro significato fisico. Esso è stato sviluppato all'interno del sistema CAMELot, che consente la definizione, la simulazione parallela, la visualizzazione, lo steering e l'analisi di modelli AC nello stesso ambiente, utilizzando una interfaccia friendly ed assicurando al contempo una notevole flessibilità. Il modello è stato validato attraverso il confronto dei risultati con schemi numerici di letteratura, mostrando prestazioni soddisfacenti anche in situazioni caratterizzate da marcate non-linearità, ed ottenendo alti valori di efficienza nell'analisi della scalabilità computazionale. Inoltre, sono state applicate alcune tecniche data-driven e di quantizzazione, mirate a trasformare il modello AC in una struttura non uniforme ed asincrona. L'uso di tali tecniche può essere sommato all'effetto del calcolo parallelo, perché esse possono determinare una significativa riduzione nel numero di interazioni locali tra automi adiacenti e limitare i cambiamenti di stato delle variabili, permettendo simulazioni più rapide senza ridurre apprezzabilmente la qualità dei risultati del modello.

La gestione della risorsa idrica nei grandi impianti per la produzione di energia elettrica: sviluppo ed uso di strumenti CFD efficienti per la sicurezza e la compatibilità ambientale.

Roberto Guandalini - CESI RICERCA Spa - ASV Dept.
Andrea Maffio - CESI RICERCA SpA - ASV Dept.

I grandi impianti per la produzione di energia elettrica, basati sull'impiego di combustibili fossili (centrali termoelettriche) oppure dell'acqua (impianti idroelettrici), interagiscono con il territorio in cui sono inseriti ponendo una serie di problematiche riguardanti la sicurezza dell'esercizio e la compatibilità ambientale degli impianti. Di particolare interesse sono da considerare le problematiche relative all'utilizzo ed alla salvaguardia della risorsa idrica, anche alla luce degli effetti dei cambiamenti climatici e/o delle potenziali importanti conseguenze sul territorio. Le centrali termoelettriche, ad esempio, a seguito delle esigenze di raffreddamento legate ai processi produttivi restituiscono nei corsi d'acqua e/o nelle aree costiere acqua a temperatura superiore a quella naturale di prelievo, ponendosi pertanto la necessità di non alterare le condizioni naturali dell'ambiente acquatico ed al contempo garantire i livelli

di produzione elettrica anche nei periodi critici (di minore disponibilità idrica e maggiori temperature) quali quelli estivi. Negli impianti idroelettrici, la presenza di grandi volumi di acqua realizzati sbarrando il corso d'acqua con una diga, può comportare problemi di sicurezza per il territorio e le infrastrutture ove l'impianto è localizzato, in concomitanza di particolari condizioni idrogeologiche (eventi di piena, instabilità dei versanti, terremoti, ecc.), malfunzionamento delle opere idrauliche e/o cedimenti strutturali. Per supportare la progettazione, gli studi di impatto ambientale e/o le azioni di mitigazione è importante disporre di strumenti adeguati in termini di costi-benefici a trattare la complessità tipica delle applicazioni ingegneristiche. CESI RICERCA, nell'ambito delle attività di ricerca per il miglioramento del sistema elettrico italiano, ha in corso da anni specifici progetti riguardanti lo sviluppo e l'uso di codici di calcolo basati sulla CFD (sia di tipo "euleriano" che impiegano una mesh di calcolo, sia di tipo "meshless" quali l'approccio Smoothed Particle Hydrodynamics – SPH), per effettuare simulazioni di problemi di idrodinamica caratterizzati da presenza di geometrie complesse, flussi multifase e fortemente variabili nel tempo, tridimensionalità. Considerando che nelle applicazioni reali occorrono un numero di variabili dell'ordine dei milioni per schematizzare in modo soddisfacente il problema e che è necessario effettuare differenti scenari di simulazione, i codici di calcolo sono stati messi a punto in ambiente parallelo, in modo da limitare l'onere computazionale, ma facendo riferimento a piattaforme di calcolo tipicamente presenti in ambito industriale. Nella memoria, verranno presentati: una sintesi delle metodologie di CFD impiegate, le principali caratteristiche dei codici di calcolo, esempi applicativi per i quali i risultati delle simulazioni sono analizzati in termini sia di accuratezza (confronto con i dati sperimentali) sia di tempi di calcolo, mettendo in luce problematiche/vantaggi derivanti dall'impiego del calcolo parallelo. Le attività di ricerca sono state finanziate dal Fondo di Ricerca per il Sistema Elettrico nell'ambito di un Accordo di Programma tra CESI RICERCA e Ministero per lo Sviluppo Economico ai sensi del Decreto M.A.P. 23 marzo 2006 e del D.L. 18 giugno 2007 n. 73.

Fluidodinamica computazionale su cluster Gigabit Ethernet: il ruolo del software di comunicazione.

Giuseppe Ciaccio - DISI - Universita' di Genova

Nel panorama del calcolo ad alte prestazioni, i calcolatori paralleli di tipo cluster rappresentano attualmente il modello piu' largamente utilizzato.

Nella classifica dei 500 calcolatori piu' veloci del mondo, i cluster basati su Gigabit Ethernet rappresentano quasi la meta' delle posizioni. Il punto di forza dei cluster basati su Gigabit Ethernet e' il costo assai basso della interconnessione tra i nodi di calcolo.

In tali calcolatori, tuttavia, la interconnessione Gigabit Ethernet costituisce anche un collo di bottiglia; la principale limitazione e' costituita dal software di comunicazione (protocollo TCP/IP, driver di rete) che impedisce a Gigabit Ethernet di raggiungere prestazioni teoricamente assai piu' elevate.

Sviluppando un software di comunicazione ad alte prestazioni denominato GAMMA (Genoa Active Massage MAchine), ottimizzato per l'uso in ambiente cluster, e' stato possibile studiare il ruolo del software di comunicazione nel livello di efficienza complessivo del sistema di calcolo.

Alcuni run di fluidodinamica computazionale sono stati condotti mediante il programma OpenFOAM in versione MPI, su un cluster Gigabit Ethernet di 20 nodi dual-Xeon. Sono stati presi in esame due scenari: un risolutore in regime stazionario per flussi turbolenti di fluidi incomprimibili, e un simulatore ai larghi vortici per flussi a volume costante. In entrambi gli scenari sono state utilizzate varie implementazioni di MPI (LAM, OpenMPI, MPICH2, e MPI/GAMMA) allo scopo di confrontarne le prestazioni a parita' di hardware.

Il confronto ha mostrato che l'uso di un software di comunicazione ottimizzato (GAMMA) puo' regalare incrementi anche notevoli di prestazione, permettendo una assai maggiore efficienza dei cluster Gigabit Ethernet nelle applicazioni di fluidodinamica.

Seconda Sessione - Algoritmi e tecniche numeriche per la CFD

Implementazione parallela di un metodo agli elementi finiti discontinui di Galerkin per la fluidodinamica su griglie poliedriche non strutturate.

Andrea Crivellini - Università degli Studi di Bergamo, Facoltà di Ingegneria, Dipartimento di Ingegneria Industriale

Nicoletta Franchina – Università degli Studi di Bergamo, Facoltà di Ingegneria, Dipartimento di Ingegneria Industriale

Antonio Ghidoni - Università degli Studi di Brescia, Facoltà di Ingegneria, Dipartimento di Ingegneria Meccanica e Industriale
Stefano Rebay - Università degli Studi di Brescia, Facoltà di Ingegneria, Dipartimento di Ingegneria Meccanica e Industriale
Pietro Tesini - Università degli Studi di Bergamo, Facoltà di Ingegneria, Dipartimento di Ingegneria Industriale

Il metodo di Galerkin Discontinuo (DG) è un metodo agli elementi finiti che approssima la soluzione all'interno degli elementi per mezzo di funzioni (principalmente polinomiali) continue a tratti senza alcuna richiesta di continuità globale. La soluzione numerica è per questo generalmente discontinua sull'interfaccia tra gli elementi. La mancanza di continuità globale dà luogo a schemi discreti caratterizzati da grande flessibilità geometrica. Questa caratteristica è alla base del crescente interesse della comunità scientifica internazionale per questa classe di metodi e per il loro utilizzo in diversi campi applicativi.

Schemi accurati di elevato ordine posso essere costruiti, senza alcuna difficoltà concettuale, sia per griglie strutturate che non strutturate, anche non conformi.

In questo lavoro è presentata un'implementazione del metodo adatta ad una discretizzazione di elevato ordine delle equazioni, stazionarie ed instazionare, comprimibili ed incomprimibili di Eulero, Navier-Stokes e RANS.

La capacità di trattare elementi poliedrici di forma generica (poligoni in 2D), conservando una approssimazione polinomiale di elevato ordine, è uno degli aspetti innovativi di questa implementazione.

Il metodo DG è spesso criticato perché considerato costoso dal punto di vista computazionale. Tuttavia lo schema è anche molto versatile, permette una facile implementazione di schemi di impliciti di soluzione e altrettanto facilmente può essere parallelizzato.

Turbulent shearless mixing.

Michele Iovieno - Dipartimento di Ingeneria Aeronautica e Spaziale, Politecnico di Torino
Daniela Tordella - Dipartimento di Ingeneria Aeronautica e Spaziale, Politecnico di Torino

The interaction of two isotropic decaying turbulences scale generates the simplest kind of inhomogeneous turbulent field. This flow is higly intermittent and exibits intense burst of vorticity and strain, resulting in non gaussian statistics.

We present numerical experiments where two time decaying isotropic fields of kinetic energies E_1 and E_2, $E_1>E_2$, initially match over a narrow region. This transition layer initially represents no more than 1/40 of the domain. The following temporal evolution produces a shearless mixing, which grows in time while the two flows decay.

It is observed that the intermittency and the depth of penetration by the eddies from the high energy region increases when the energy and lengthscale gradients are concordant and decreases when they are opposite.

Moreover, when the integral scale is kept uniform across the mixing layer, this numerical experiment shows that the presence of a turbulent energy gradient is sufficient for the appearance of intermittency and that during the mixing process the pressure transport is not negligible with respect to the turbulent velocity transport.

Inermittency increases with the energy ratio E_1/E_2 reaches a maximum in the limit for $E_1/E_2 \rightarrow\infty $, which corresponds to the diffusion of a turbulent flow into a region with still fluid.

These findings may open the way to the hypothesis that the presence of a gradient of turbulent energy is the minimal requirement for Gaussian departure in turbulence.

A Lagrangian finite element approach to the simulation of fluid-structure interaction.

Massimiliano Cremonesi - Dipartimento di Ingegneria Strutturale, Politecnico di Milano
Attilio Frangi - Dipartimento di Ingegneria Strutturale, Politecnico di Milano
Umberto Perego - Dipartimento di Ingegneria Strutturale, Politecnico di Milano

The equations of motion for a Newtonian fluid are usually formulated in Eulerian form. In many situations with a fixed domain or control volume this formulation allows to solve the problem conveniently. However, there are situations, for instance free-surface flows or flows with breaking waves, in which the Eulerian formulation is not convenient in view of the difficulty in defining the boundary conditions at material surfaces. The same difficulties arise when the fluid flow is coupled with a solid which undergoes large deformations.

The treatment of free surfaces and solid-fluid interfaces is better dealt with in a Lagrangian framework where boundary and interface conditions can be explicitly enforced. ALE methods (Arbitrarian Lagrangian Eulerian methods), often used for these type of problems, represent an intermediate approach which proves useful in many circumstances. In the Lagrangian formulation, the Navier-Stokes equations are formulated using reference coordinates which are updated at each time-step of the time-integration procedure [Radovitzky and Ortiz, 1998]. The main difficulty in the Lagrangian approach is represented by the distortion of the mesh. If a fixed finite element mesh is used and the position of element nodes is updated as a consequence of the fluid flow, very soon the element distortion becomes excessive. A remedy which allows to avoid these distortions consists of systematically remeshing the volume of the problem.

Using a Lagrangian formulation the nonlinear convective term, typical of Eulerian formulations, disappears. However, when remeshing is introduced within the time-step solution algorithm, a new source of nonlinearity appears due to the fact that the discretized equations are written using matrix operators based on an unknown geometry. In the present work, we develop a Lagrangian finite element method for the analysis of incompressible Newtonian fluid flows based on a continuous remeshing, in the spirit of the so called Particle finite element method [Idelsohn et al., 2006]. The key feature of this method is that not only the fundamental physical properties, such as density and viscosity, but also velocity and pressure, are assigned to particles, which are identified with element nodes. In this way, the element nodes represent a real part of the physical domain. To update the position of the particles at every time-step, velocity and pressure are computed solving the Lagrangian version of Navier-Stokes equations, discretized by finite elements.

An additional difficulty connected to the particle finite element approach, is represented by the fact that the fluid domain where the Navier-Stokes equations are to be solved is defined by the positions of the particles. At each time step the particles change their positions, and consequently the domain changes. Particles belonging to the boundary at a time-step, may not be on the boundary at subsequent time-steps. This implies that a method to identify the external boundary is required to define the current integration domain and to impose the boundary conditions of the differential problem. To this purpose, use is made here of the so called "alpha-shape method" proposed in [Idelsohn et al., 2006]}.

Since both fluid and structure are formulated in Lagrangian framework, the proposed implementation is particularly suitable for fluid-structure interaction problems. Simple two-dimensional examples of fluid-structure interaction are presented to test the potential of the proposed method.

References:

[1] (Radovitzky and Ortiz, 1998) Radovitzky R., Ortiz M., Lagrangian finite element analysis of newtonian fluid flows, Int. J. Numer. Methods Engrg. 43 (1998) 607-619

[2] (Idelsohn et al.,2006) Idelsohn S. R., Oñate E., Del Pin F., Calvo N., Fluid-Structure interaction using the particle finite element method, Comput. Methods Appl. Mech. Engrg. 195 (2006) 2100-2123

Active control and drag reduction in turbulent wall flows.

Fulvio Martinelli - Dipartimento di Ingegneria Aerospaziale, Politecnico di Milano
Maurizio Quadrio - Dipartimento di Ingegneria Aerospaziale, Politecnico di Milano
Paolo Luchini - Dipartimento di Meccanica, Università di Salerno

Active control of turbulent wall flows is an emerging discipline, which is gaining more and more attention from the scientific community; active control techniques look in fact promising in many areas such as drag reduction, separation delay, noise suppression and mixing enhancement.

In particular, new encouraging results have been recently obtained by using modern linear control theory in the design of active feedback controllers, aimed at reducing skin friction drag in turbulent channel flows by means of distributed wall blowing and suction.

These results have been obtained via computationally intensive direct numerical simulations and solution of high-dimensional optimal control problems, for Reynolds numbers as low as Re=1450.

In this work, after an introductory overview of the most important achievements in this area, we focus on the description of the main results obtained by our group. In particular, the Reynolds number effect on the performance of optimal controllers has been investigated, via well-resolved direct numerical simulations of the actively controlled turbulent channel flow at about Re=10000.